RESEARCH
Indian Journal of Engineering, Volume 2, Number 4, February 2013
Indian Journal of Engineering
ISSN 2319 – 7757 EISSN 2319 – 7765

Analysis of an automotive oil filter housing assembly using finite element method

Gopinathan R[1], Lawrence P[2], Anitha Y[3], Santhanakrishnan S[4], Ramu S[5]

1. Department of Mechanical Engg, C.Abdul Hakeem College of Engineering, Vellore, India
2. Department of Mechanical Engg, Podhigai College of Engineering & Technology, Tirupattur, India
3. Department of Mechanical Engg, Global Institute of Engineering & Technology, Vellore, India
4. Department of Mechanical Engg, Shri Sapthagiri Institute of Technology, Ocheri, Vellore, India
5. Department of Mechanical Engg, Shri Ramana Maharishi College of Engineering, Cheyyar, India

Received 06 December; accepted 03 January; published online 01 February; printed 16 February 2013

ABSTRACT
The oil filter housing assembly is the part of the lubrication system of an automobile and since it is located near to the engine, the vibrations are enormous. So it is more appropriate to do the analysis on the oil Filter Housing assembly of an automobile (car). This investigation effectively deals with the meshing of the oil filter housing components. It is meshed with tetrahedral elements using Hypermesh 8.0. The meshing is done with the given quality criterias and is converted into Ansys template file (*.prp file). Then the analysis is carried out in Ansys10.0. The analysis part includes modal and Frequency response analysis. The modal analysis aims in finding out the natural frequencies of the system and also to find whether the meshing of the component is correct i.e. nodal connectivity between parts exists or not. The Frequency Response Analysis aims in finding out whether the system has the chances of undergoing any resonance due to the forced frequencies.

Keywords: Oil filter housing assembly, frequency response analysis, modal analysis.

1. INTRODUCTION
Nearly all multi cylinder engines used in automotive, construction, and material-handling equipment use a liquid-cooled system. A simple lubricating system consists of a radiator, coolant pump, piping, fan, thermostat, and a system of water jackets and passages in the cylinder head and block through which the coolant circulates. Some vehicles are equipped with a coolant distribution tube inside the cooling passages that directs additional coolant to the points where temperatures are highest. Cooling of the engine parts is accomplished by keeping the coolant circulating and in contact with the metal surfaces to be cooled (Kirpal Singh, 2011). The operation of a liquid-cooled system is as follows: The amount of engine heat that must be removed by the cooling system is much greater than is generally realized. To handle this heat load, it may be necessary for the cooling system in some engine to circulate 18,000 to 45,000 LITERS of coolant per hour (Kirpal Singh, 2011).

2. THE FINITE ELEMENT METHOD
If the geometry of the given system is complex, it seems that the domain can be represented as an assembly of geometrical simple sub domains for which the construction of approximate function s become feasible. The FEM is based on these ideas. In the FEM, a given domain is discredited by a collection of geometrical simple sub domains called Finite Elements and for each element of the collection, the element equations are derived using any one of the variational principles. The approximations of field variables, now on each element, are systematically generated. The elements are connected together by imposing the continuity of field variables not only at their nodes but also across the inter element boundaries (Dalessandro, 2005).

2.1. Advantages of the Finite Element Method
In initiating the prediction of displacements, stresses, vibration frequencies, buckling loads, etc. for a given structure or machine, the analyst must first derive the governing equation. A basic difficulty in this approach, quite apart from the solvability of the derived equations is the ability of these equations to represent the design conditions. Complexity in geometry, applied loads, support conditions and material properties enter into this condition. The FEM brings a number of special advantages to thermal analysis. A consistent methodology of finite element heat transfer is available for the calculation of temperature distribution in solids and structures. It is possible to use same general-purpose FEA program to the calculation of both temperature distributions and thermal stresses.

Table 1 Quality Criteria's for 2D elements

Type	Limit	No. Of Elements Failed	% Of Elements Failed
Warpage	> 5	0	0
Aspect Ratio	> 5	0	0
Skew	> 60	42	0
Jacobian	< 0.7	0	0
Minimum Angle	< 15	0	0
Maximum Angle	> 135	9	0

3. FINITE ELEMENT MODELING USING HYPER MESH
3.1. 2D Mesh
The different parts of the assembly kept in different collectors, are meshed separately for each part. First the component is meshed with 2-D tria3 elements. The required quality criteria's for the mesh is maintained throughout. After meshing, the element connectivity is checked. Equivalence, if any present is given with tolerance 0.01. Then the qualities of the elements are been checked. The following table gives the quality criteria

Gopinathan et al.
Analysis of an automotive oil filter housing assembly using finite element method,
Indian journal of engineering, 2013, 2(4), 17-20,

Table 2 Quality Criteria's for 3D elements			
Type	**Limit**	**No. Of Elements Failed**	**% Of Elements Failed**
Skew	> 60	371	0
Tet collapse	< 0.15	0	0
Minimum Angle	< 15	100	0
Maximum Angle	> 135	54	0

are maintained for criteria3 elements. Total number of tria3 elements created in the model is 85,409. The nodal connectivity between the parts is given in the 2D mesh itself. For giving this connectivity, the elements of one of the mating parts are projected on the other. These elements are then used for the meshing of the mating component.

3.2. 3D Mesh

After correcting the failed elements and free edges of the 2D mesh, the 3D tetrahedral mesh is created using tetra4 element. Since the nodal connectivity is given in 2D elements, 3D elements will also have connectivity. The elements created are then checked with the quality criteria's shown in the below table. Total number of 3D tetra4 elements created is 1,62,423 & No of nodes found are 48,382. The meshed components of the assembly are shown in Fig.1 to Fig.5

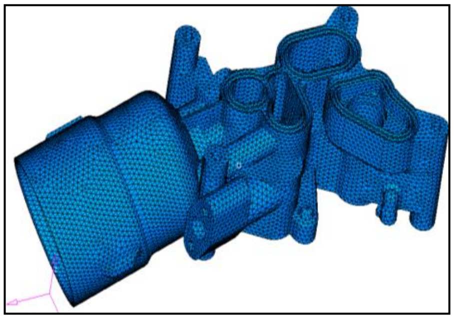

Figure 1
Main Housing

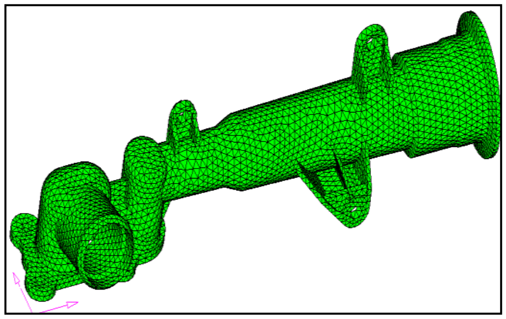

Figure 2
Secondary Filter

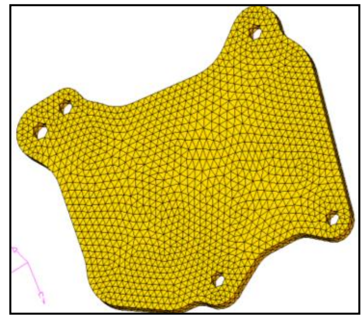

Figure 3
Base Plate

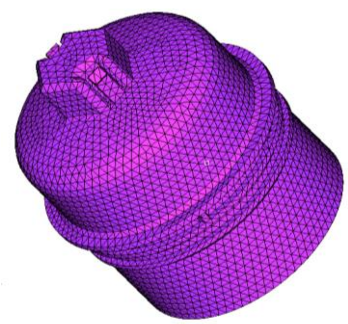

Figure 4
Primary oil filter cap

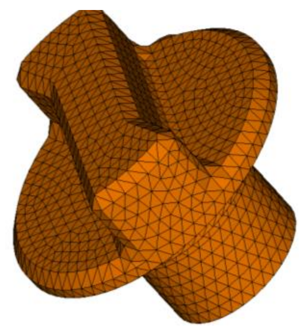

Figure 5
Secondary oil filter cap

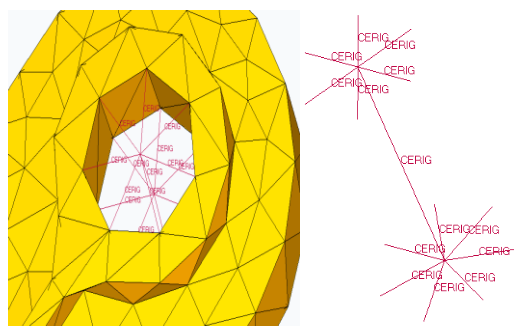

Figure 6
Rigids

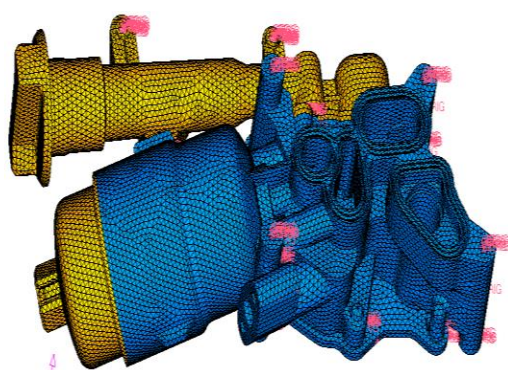

Figure 7
Assembled Finite Element Model

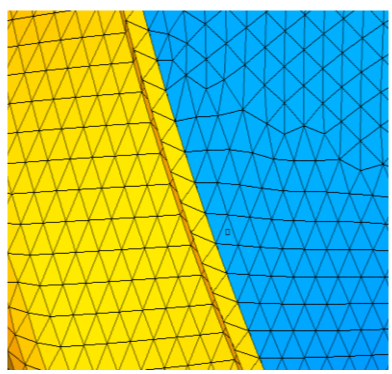

Figure 8
Nodal Connectivity between Parts

3.3. Creation of Rigids

After meshing all the parts with the nodal connectivity between the parts, the bolted locations of the assembly are given rigid elements. First temporary nodes are created at the center of the hole, both in the top and bottom surfaces of the hole. Then the Rigids are created by selecting the center node as the independent node (master node) and the remaining nodes as the dependent node (slave node). The element type for the rigids is given as CERIG.

4. FINITE ELEMENT ANALYSIS

The component is made up of two materials. One is of Aluminium and the other is plastic.

4.1. Material Property

Table 3 Aluminium Material Property		
Property	**Aluminium**	**Plastic**
Mass Density (ρ)	2.74e-6 kg/mm3	1.3 gm/cm3
Young's Modulus (E)	71705 mPa	3450 mPa
Poisson's Ratio	0.33	0.3

Gopinathan et al.
Analysis of an automotive oil filter housing assembly using finite element method,
Indian journal of engineering, 2013, 2(4), 17-20,

Table 5 Natural Frequency Values

SET	TIME/FREQ	LOAD STEP	SUBSTEP
1	352.34	1	1
2	456.31	1	2
3	1096.4	1	3
4	1222.2	1	4
5	1427.5	1	5
6	1447.5	1	6
7	1892.7	1	7
8	2050.7	1	8
9	2057.9	1	9
10	2065.2	1	10
11	2097.1	1	11
12	2282.3	1	12
13	2438.8	1	13
14	2744.1	1	14
15	2790.4	1	15

Table 4 Free- Free Modal frequency values

Extracted Modes	Natural Frequency (Hz)
1	0
2	0
3	0
4	0
5	0
6	0
7	0.53195E-03
8	0.98903E-03
9	0.25534E-02
10	0.36218E-02
11	37669E-02
12	0.71921E-02
13	83.369
14	85.776
15	439.10

4.2. Mode Extraction Method Used

The Block Lanczos method is used for large symmetric eigen value problems. We can use this method for the same types of problems for which we use the subspace method, but we achieve a faster convergence rate. The Block Lanczos method uses the sparse matrix solver, overriding any solver specified via the EQSLV command.

4.3. Free - Free Modal Analysis

First modal analysis on the assembly is made without applying any boundary conditions. This is done to know whether nodal connectivity is there between the parts. This is ensured by seeing the first six set values, which will be zero. No damping effects included. The following table gives the natural frequencies of the assembly with out any boundary condition. The first 15 modes are been extracted and the values are given in Table 4.

4.4. Fixed Modal Analysis

After completing the free-free modal analysis, the forced model analysis is done. Here the boundary conditions are given. Bolted holes are arrested in all degree of freedom. And an acceleration of 10 g is been provided. And then the modal analysis is done. The following table gives the different frequencies of the system. Since the working range is from 40 Hz, the frequencies are also obtained above 40Hz. The results and the mode shapes obtained are given in the following pages. The different mode shapes obtained are given in Figure 9 and Figure 10

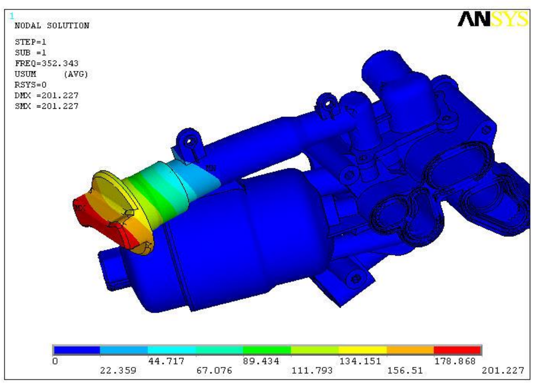

Figure 9

1st Mode shape (Frequency 352.343Hz)

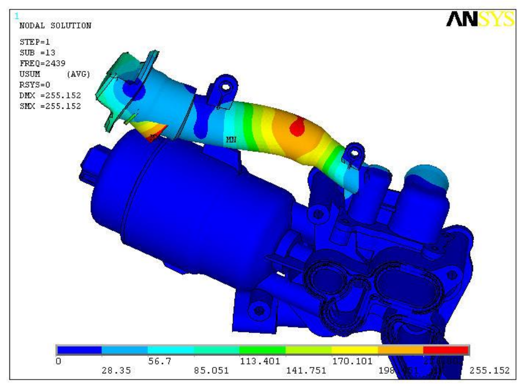

Figure 10

13th Mode shape (Frequency 2439 Hz)

5. DISCUSSION OF RESULTS

In the free – free modal analysis the natural frequencies of the assembly is found out. From the results it is concluded that the connectivity of the FE model is correct since the first six natural frequency values are found as zero. In the Fixed modal analysis, again the natural frequencies of the system in the fixed condition are obtained. These results are obtained for the further analysis of the FE model that is Frequency response analysis.

5.1. Harmonic Response Analysis

Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies. For a harmonic analysis, Both Young's modulus (EX) (or stiffness in some form) and density (DENS) (or mass in some form) must be defined. Material properties may be linear, isotropic or orthotropic, and constant or temperature-dependent. Nonlinear material properties, if any, are ignored. On the same model with the same material properties, the harmonic frequency response analysis is been done with the Full Method. The boundary conditions are applied with all bolted holes arrested in all degree of freedom and an acceleration of 10 g is been provided. In the load step options the frequency range is given as 40 – 500 Hz and the number of sub steps is given as 15. The following are the results obtained for the harmonic analysis. The graphs are plotted between the frequency and amplitude and another between frequency and Von mises stress. In the Graph (Figure 11), there are two peaks obtained. This indicates the coincidence of the natural and forced frequencies. The Peak amplitudes obtained are at 348 Hz and at 470 Hz. The maximum amplitude obtained is 0.6 mm. In the graph obtained (gig 12) we are getting two peak amplitudes at the frequencies 347 and 465 Hz. The maximum displacement obtained is 0.6 mm. The maximum stress obtained is 3.9 e-2 N/mm2. These two maximum frequencies are approximately matching with that of the natural frequencies 352 Hz and 456 Hz and this is the

Gopinathan et al.
Analysis of an automotive oil filter housing assembly using finite element method,
Indian journal of engineering, 2013, 2(4), 17-20,

reason for the sudden peak in amplitude. So when the forced frequency of a system matches with that of the natural frequency, then there is a sure chance of resonance in the system. So if the system works in this given range then, definitely the system is going to fail due to resonance.

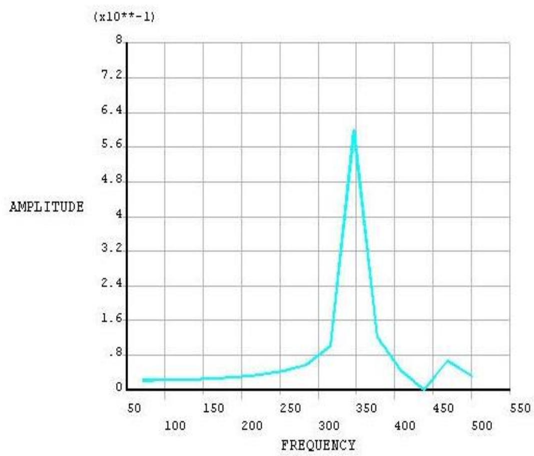

Figure 11
Frequency Vs Amplitude Plot

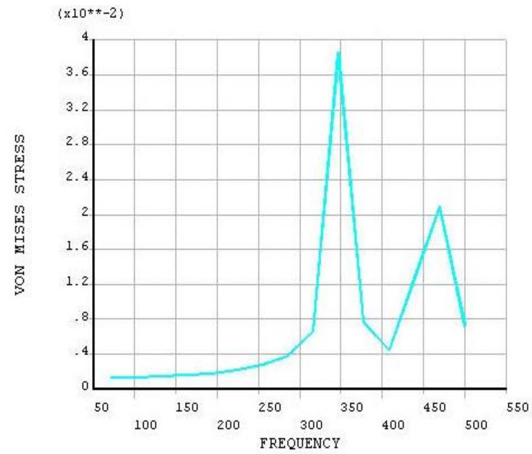

Figure 12
Frequency Vs Von Mises Stress Plot

6. CONCLUSION

This paper has effectively dealt with the meshing and analysis of an oil filter housing assembly. The model is meshed successfully in Hypermesh with tetrahedral elements with the required quality criterias. After meshing, the finite element model is converted into Ansys template and then opened in Ansys. In the modal analysis of the assembly, it is ensured that the meshing of the components is correct. The natural frequencies of the system are also found out. In the harmonic response analysis it is verified that whether the designed system will undergo any resonance or not. From the results it is found that the system will undergo resonance at the frequency of 352 Hz. So the system will definitely fail if it works in the given frequency range of 40 – 500 Hz. To avoid this resonance occurring in the system, either the working range of the system should be changed (within 350 Hz) or the material used for the component should be changed.

REFERENCES

1. Kirpal Singh. Automobile Engineering, Standard Publishers Distributors, 2011
2. ANSYS Release 7.0 Documentation, ANSYS help
3. Newland DE. An introduction to random vibrations, spectrum and wavelet-analysis, New York, Jon Wiley, USA, 1993
4. Gere, Timoshenko. Mechanics of Materials, 1st Edition, CBS Publishers and Distributors
5. Dalessandro L. Finite-element analysis of the frequency response of a metallic cantilever coupled with piezoelectric transducer, IEEE Transactions on Instrumentation and Measurement, 2005, 54, 1881-1890
6. Wang FH. Using the vibration frequency response analysis method to detect the winding deformation of power transformer, IEEE Power and Energy Society General Meeting, 1-6
7. globalsecurity.org (23/11/2005)
8. sweethavenpublishing.org/automotive technology17.htm (23/11/2005)
9. ltair.com/software/hm.htm, 13/12/2005

Gopinathan et al.
Analysis of an automotive oil filter housing assembly using finite element method,
Indian journal of engineering, 2013, 2(4), 17-20,